Printed Circuit Board
(PCB)

Helena Sarmento

Outline

- PCB design for high frequencies
- PCB board
- PCB design: units, grids, track/gap, pad, via, polygon, clearance

PCB design for high frequencies

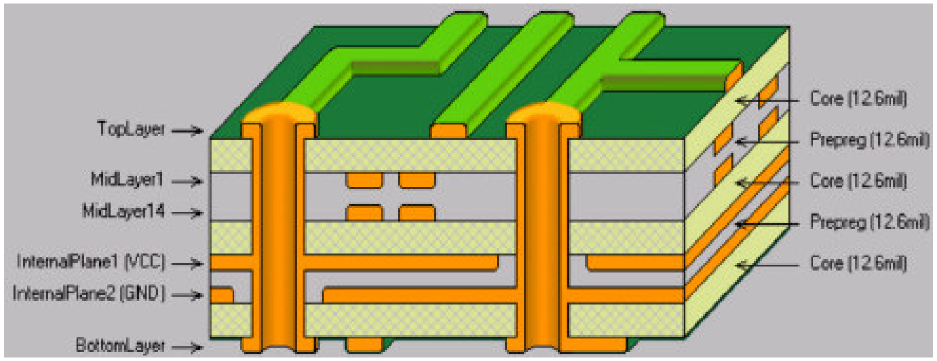
- PCB design is part of the electronic design process
- In many designs, the PCB layout may make or break the operation and electrical performance of the design: PCB traces have resistance, inductance and capacitance.
- In high speed digital and RF (analog), PCB design is critical.

PCB board

- Two basic parts:
  - Substrate: structure that physically holds circuit components and printed wires and provides isolation between conductive parts
  - and copper (wire/traces)
- Substrate
  - Teflon, ceramics, special polymers, fiber-glass-epoxy resin
  - FR-4 is the most common dielectric substrate used in the construction of PCBs.
  - FR-4 is a laminate composed of glass and epoxy that meets a specific standard for fire-retardance (FR).

PCB board

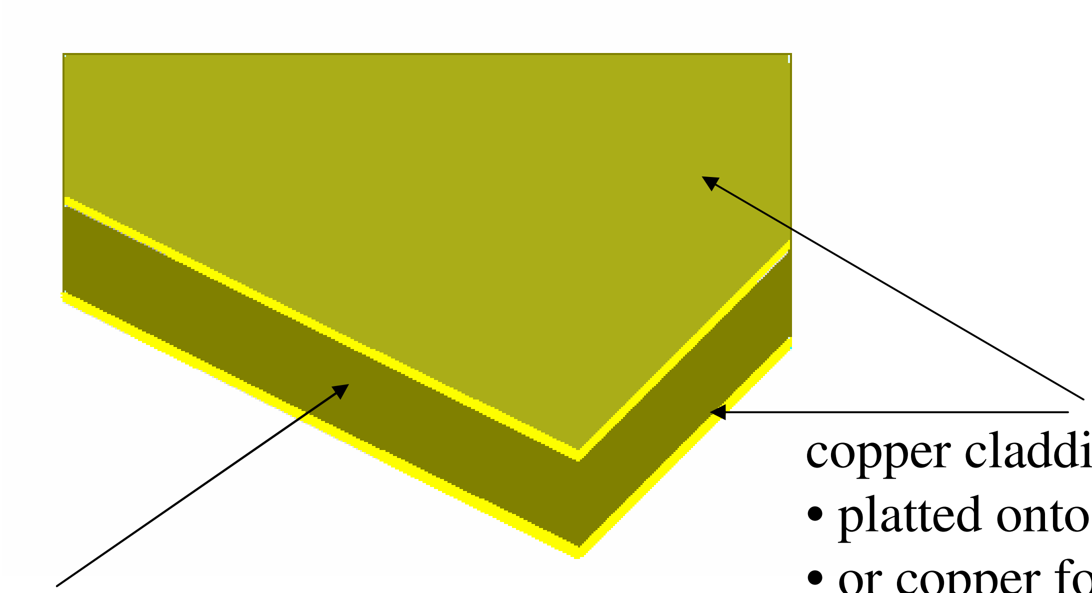
- Multilayer board:
  - cores (substrate with copper clads)
  - prepreg (**pre**-im**preg**nated)



* Figure from: Steve Rose, Printed Circuit Board Design, Development and Fabrication Process, PCB3D.COM

PCB Board

- Core: double-sided copper clad FR4 substrate



copper cladding:
- platted onto the substrate
- or copper foil glued to the substrate

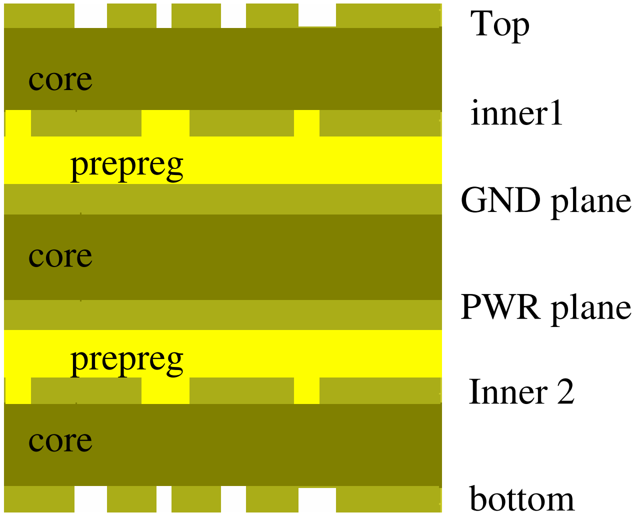
FR4 substrate

* Figure from: Kraig Mitzner, Complete PCB design using OrCAD Capture and Layout

PCB Board

Multilayer boards

- Cores glued together with one or more sheets of partially uncured fiber glass epoxy (prepreg or B-stage laminate).
- Prepreg will cure (i.e. harden) when heated and pressed
- Assembling cores: cores patterned, cores aligned (registration), entire assembly is fully cured in a heated press

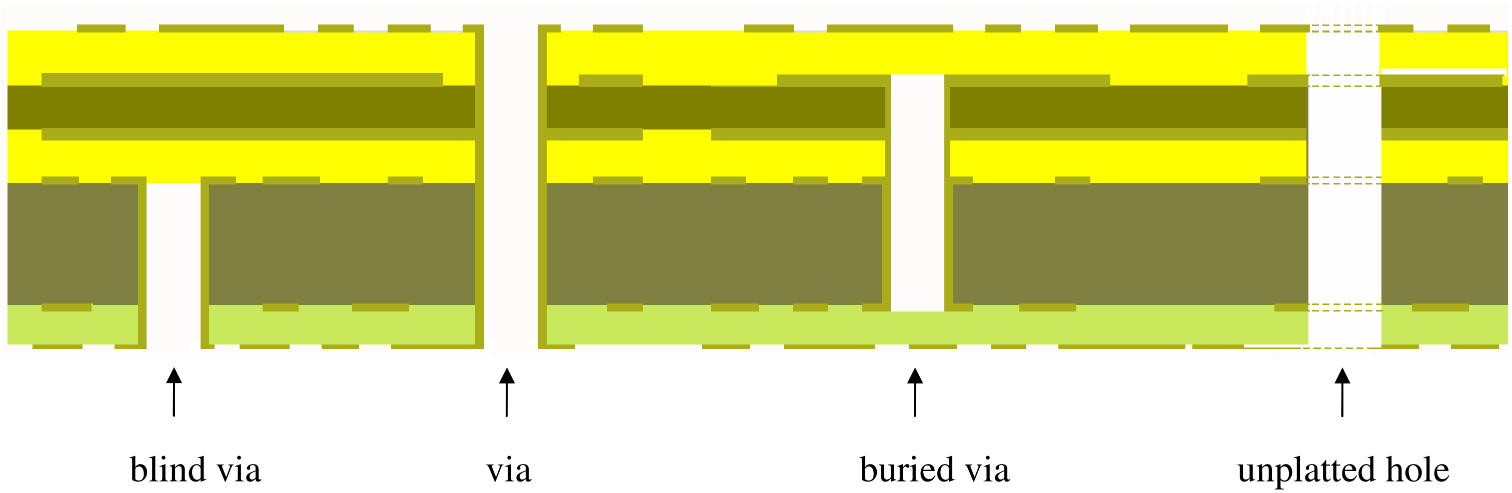


3 double-sided cores bonded together by two prepreg layers
6 layers board with 4 routing layers and two plane layers

* Figure from: Kraig Mitzner, Complete PCB design using OrCAD Capture and Layout

PCB Board

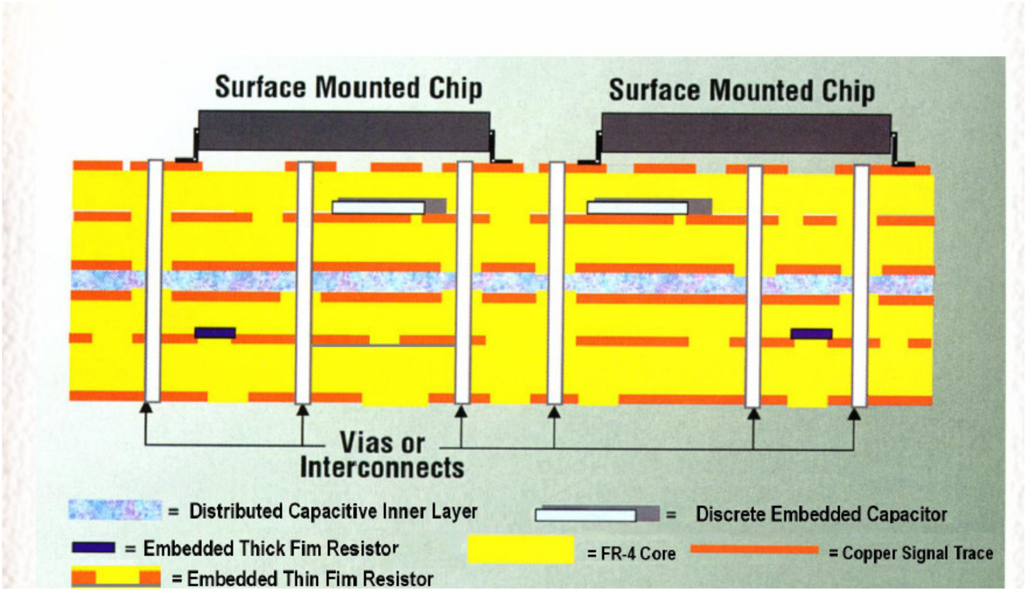
- Layers built layer by layer to build vias, blind vias and buried vias



* Figure from: Kraig Mitzner, Complete PCB design using OrCAD Capture and Layout

PCB Board

- EPC (Embedded passive components)
- Passive components placed between the interconnecting structure of board: existing passive components, specially-made thin passive components, and make film devices through printing or other processes.


Surface Mounted Chip
Surface Mounted Chip
Vias or Interconnects
= Distributed Capacitive Inner Layer
= Discrete Embedded Capacitor
= Embedded Thick Fim Resistor
= FR-4 Core
= Copper Signal Trace
= Embedded Thin Fim Resistor

* Figure from: www.ansoft.com/thewave/New_High_performance_cost_effective_PCBs.pdf

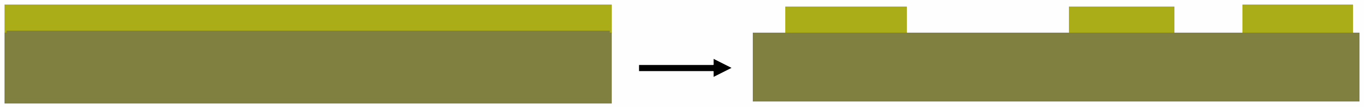
PCB Board

- How printed circuit boards are manufactured
  - Film Generation
  - Shear Raw Material
  - Drill Holes
  - Electroless Copper
  - Apply Image
  - Pattern Plate
  - Strip & Etch
  - Solder mask
  - Solder coat
  - Nomenclature
  - Fabrication

http://www.pcbpro.com/pcb-tutorial.php

PCB Board

- Copper patterning
  - produced by selectively removing copper cladding based on a digital map generated by CAD software



  - Acid etching (large production)
  - Mechanical milling (small production)

PCB Board

- Copper patterning

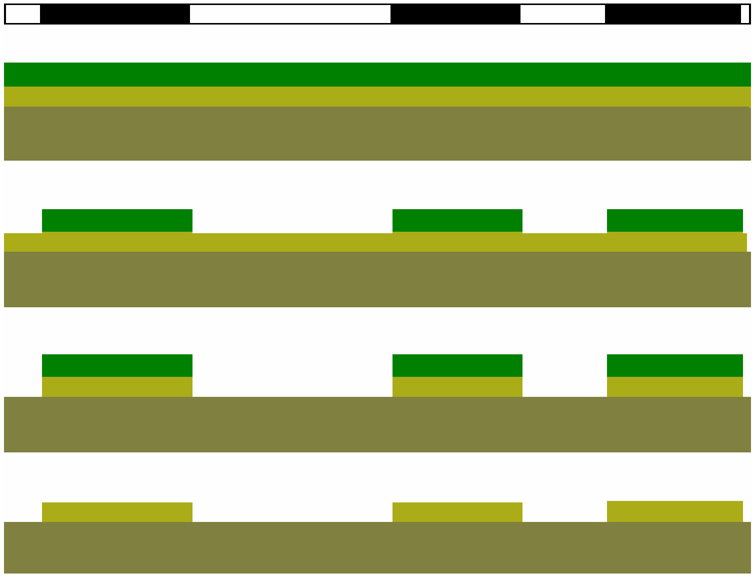


Photo mask placed
over photoresist during exposure

Copper left exposed in regions
to be removed

Exposed copper etched away

Photoresist then removed

- Multilayer: several signal layers, power and ground planes

PCB Board

- After copper patterning and cores and prepreg assembly:
  - holes are drilled for through-hole components and vias
  - electrical connections between layers are plated
  - a thin polymer layer is applied on the top and bottom of the board
  - markings are placed on the board to identify components
- There are other layers besides the copper layers
  - Drill layer
  - Soldes mask
  - Silkscreen

PCB Board

- Silk screen (top/bottom): the layer that contains your component outlines, designators and free text.

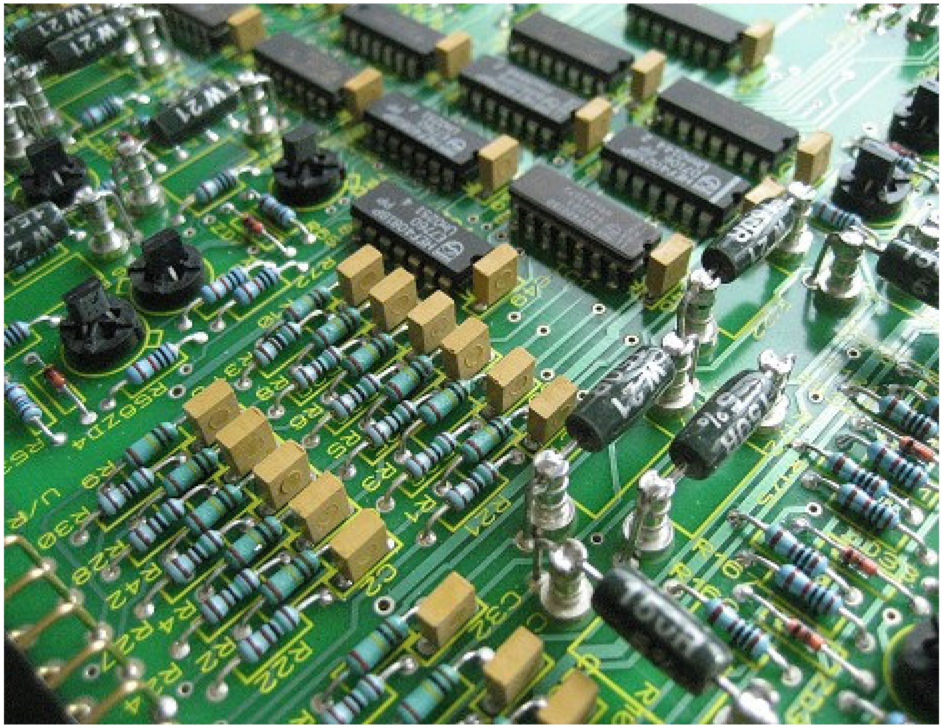


Figure from: http://www.pcb-manufacturers.co.uk/pcb-production-examples-e.html

PCB Board

- Additional layers for assembly purposes:
  - Solder paste (top/bottom): for soldering purposes. The board is coated with a mask of solder paste over the pads Then each component is placed, and is sometimes held in place by an adhesive. The entire board is then loaded into an oven and "baked". The solder paste melts (reflows) on the pads and component leads to make the joint.
  - Assembly (top/bottom): for automatic placement of components

Figure from: http://www.pcb-manufacturers.co.uk/pcb-production-examples-e.html

PCB Board

- Solder mask (top/bottom): : thin polymer used to protect the board and circuitry during the assembly and packaging operations. It prevents solder bridges between adjacent pads and traces during the wave soldering process. Typically, it covers everything except pads and vias.

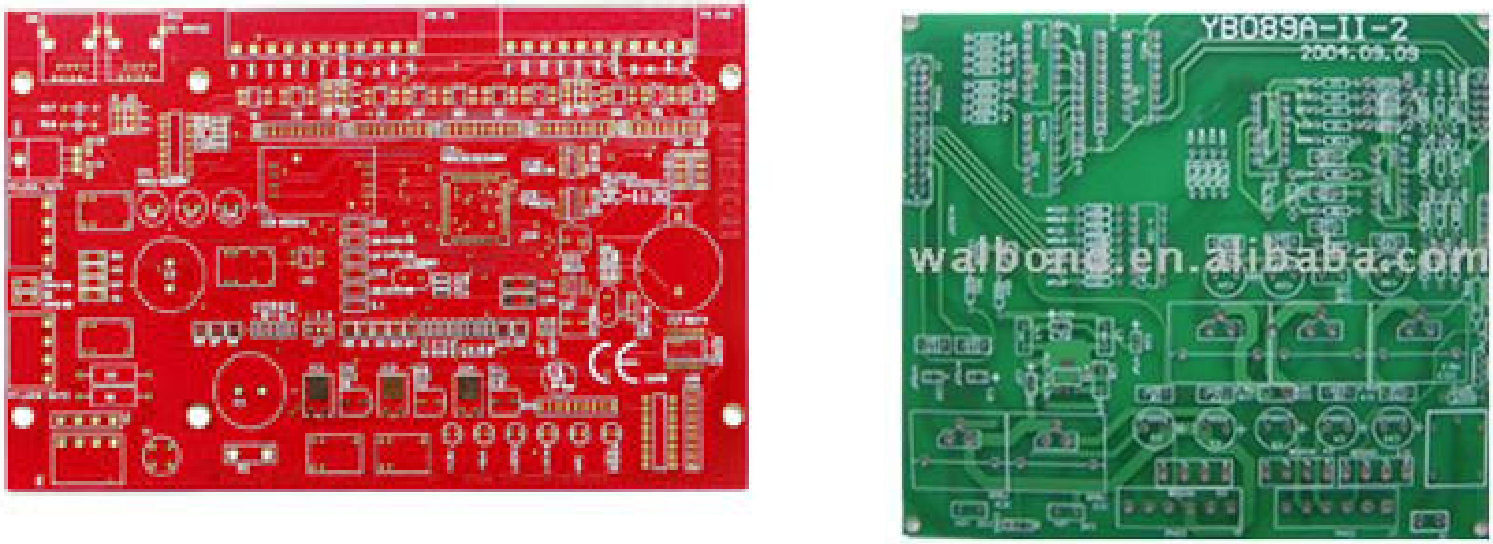


Figure from: http://www.pcb-manufacturers.co.uk/pcb-production-examples-e.html

PCB design

- Units:
  - inch: mil (1/1000 of an inch)

    mainly used for design and layout requirements: tracks, pads, spacing, grids
  - mm

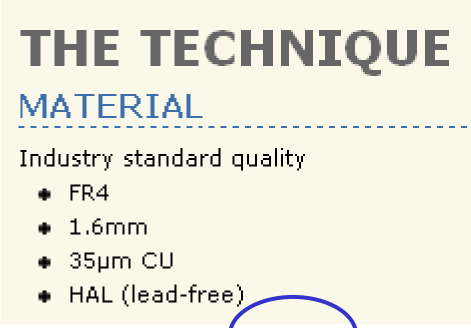
    mainly used for mechanical and manufacturing requirements: hole sizes and board dimensions
  - *Thou* (thousands of mils)

1000 mil = 25.4 mm
50 mil = 1.28mm
40 mil approx 1.00 mm
10 mil approx 0.25 mm
4 mil approx 0.10 mm

PCB design

- Examples

  www.pcb-pool.com

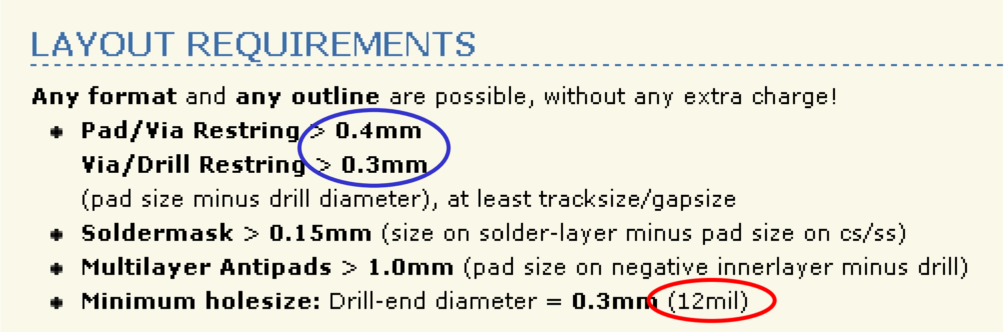


THE TECHNIQUE

MATERIAL

Industry standard quality

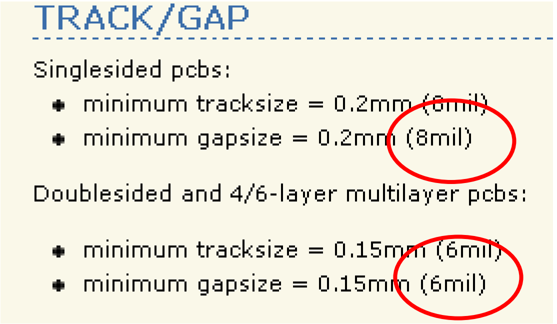
- FR4
- 1.6mm
- 35µm CU
- HAL (lead-free)



LAYOUT REQUIREMENTS

Any format and **any outline** are possible, without any extra charge!

- **Pad/Via Restring > 0.4mm**
  Via/Drill Restring > 0.3mm
  (pad size minus drill diameter), at least tracksize/gapsize
- **Soldermask > 0.15mm** (size on solder-layer minus pad size on cs/ss)
- **Multilayer Antipads > 1.0mm** (pad size on negative innerlayer minus drill)
- **Minimum holesize:** Drill-end diameter = **0.3mm** (12mil)



TRACK/GAP

Singlesided pcbs:

- minimum tracksize = 0.2mm (8mil)
- minimum gapsize = 0.2mm (8mil)

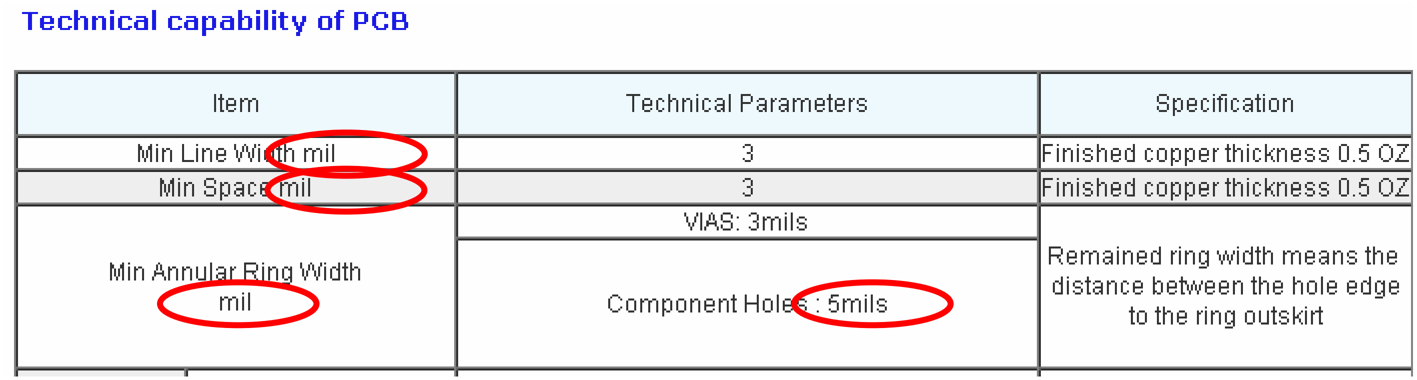
Doublesided and 4/6-layer multilayer pcbs:

- minimum tracksize = 0.15mm (6mil)
- minimum gapsize = 0.15mm (6mil)

PCB design

- Examples

http://www.pcbcore.com



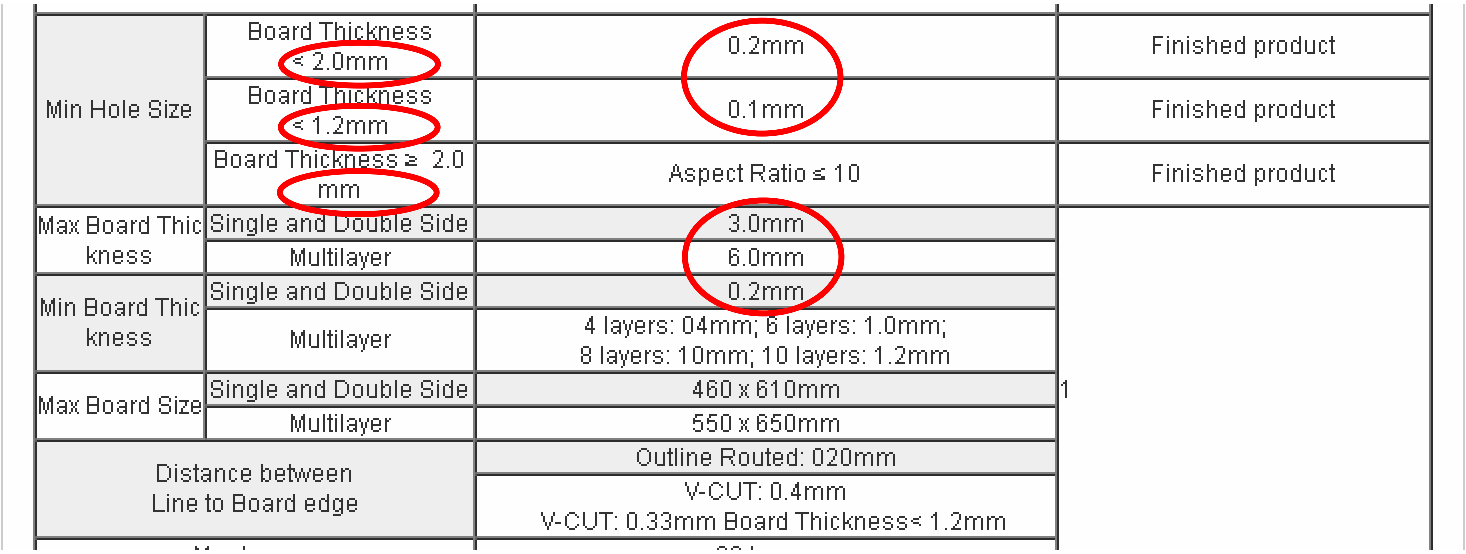
Technical capability of PCB

Item	Technical Parameters	Specification
Min Line Width mil	3	Finished copper thickness 0.5 OZ
Min Space mil	3	Finished copper thickness 0.5 OZ
Min Annular Ring Width mil	VIAS: 3mils Component Holes : 5mils	Remained ring width means the distance between the hole edge to the ring outskirt

PCB design

- Examples

http://www.pcbcore.com



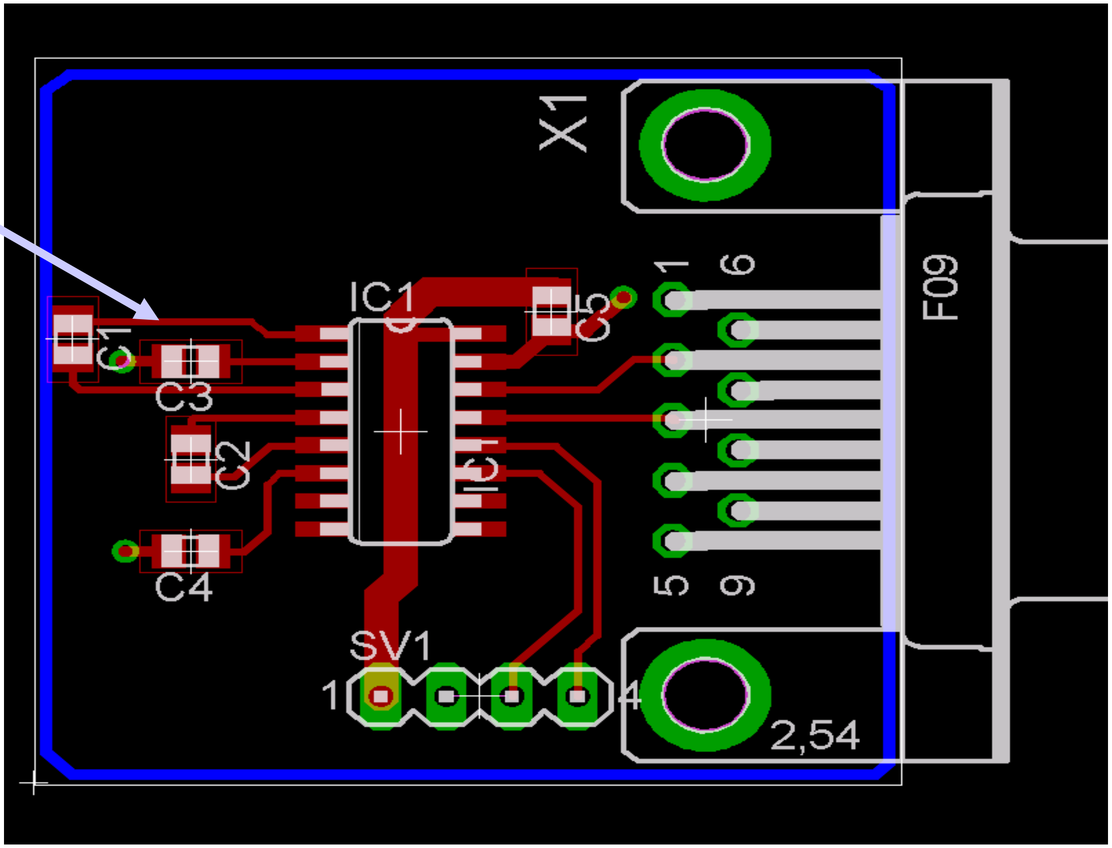
Min Hole Size	Board Thickness < 2.0mm	0.2mm	Finished product
	Board Thickness < 1.2mm	0.1mm	Finished product
	Board Thickness ≥ 2.0 mm	Aspect Ratio ≤ 10	Finished product
Max Board Thickness	Single and Double Side	3.0mm	
	Multilayer	6.0mm	
Min Board Thickness	Single and Double Side	0.2mm	
	Multilayer	4 layers: 04mm; 6 layers: 1.0mm; 8 layers: 10mm; 10 layers: 1.2mm	
Max Board Size	Single and Double Side	460 x 610mm	1
	Multilayer	550 x 650mm	
Distance between Line to Board edge		Outline Routed: 020mm	
		V-CUT: 0.4mm V-CUT: 0.33mm Board Thickness< 1.2mm	

PCB design

- Grids
  - Snap grids:
    - components and tracks will "snap" into fixed grid positions.
    - will keep components neat and symmetrical
    - makes future editing, dragging, movement and alignment of tracks, components and blocks of components easier as layout grows in size and complexity.
    - Examples: 100 mils, 50 mils, 25 mils depending on the design.
  - Component grids:
    - Like snap grids, but for component movement only.

PCB design

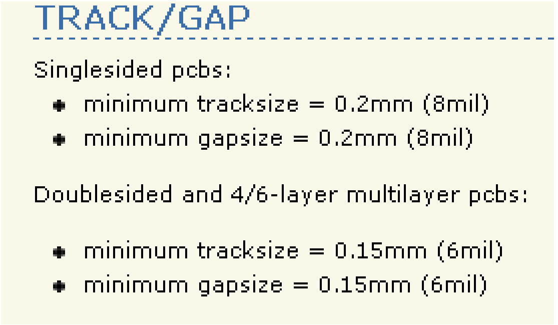
Tracks



* Figure from: www.ece.msstate.edu/courses/ece4512/des1_syllabus/current/PCB_Layout_Tips.ppt

PCB design

- Tracks
  - copper connections between two points on the PCB.
  - designs require a mixture of track sizes.
  - track size depend upon the electrical requirements of the design, the routing space and clearance available, and designer preference.
  - lower limit of the track width depend upon the *track/gap* resolution of the PCB manufacturer.
  - Examples:
    - PCB-pool:



TRACK/GAP

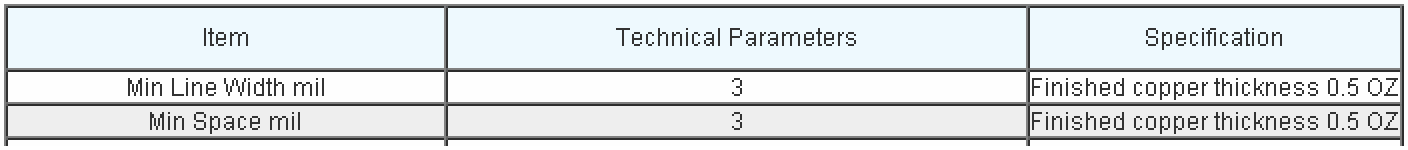
Singlesided pcbs:

- minimum tracksize = 0.2mm (8mil)
- minimum gapsize = 0.2mm (8mil)

Doublesided and 4/6-layer multilayer pcbs:

- minimum tracksize = 0.15mm (6mil)
- minimum gapsize = 0.15mm (6mil)

    - PCBCore:



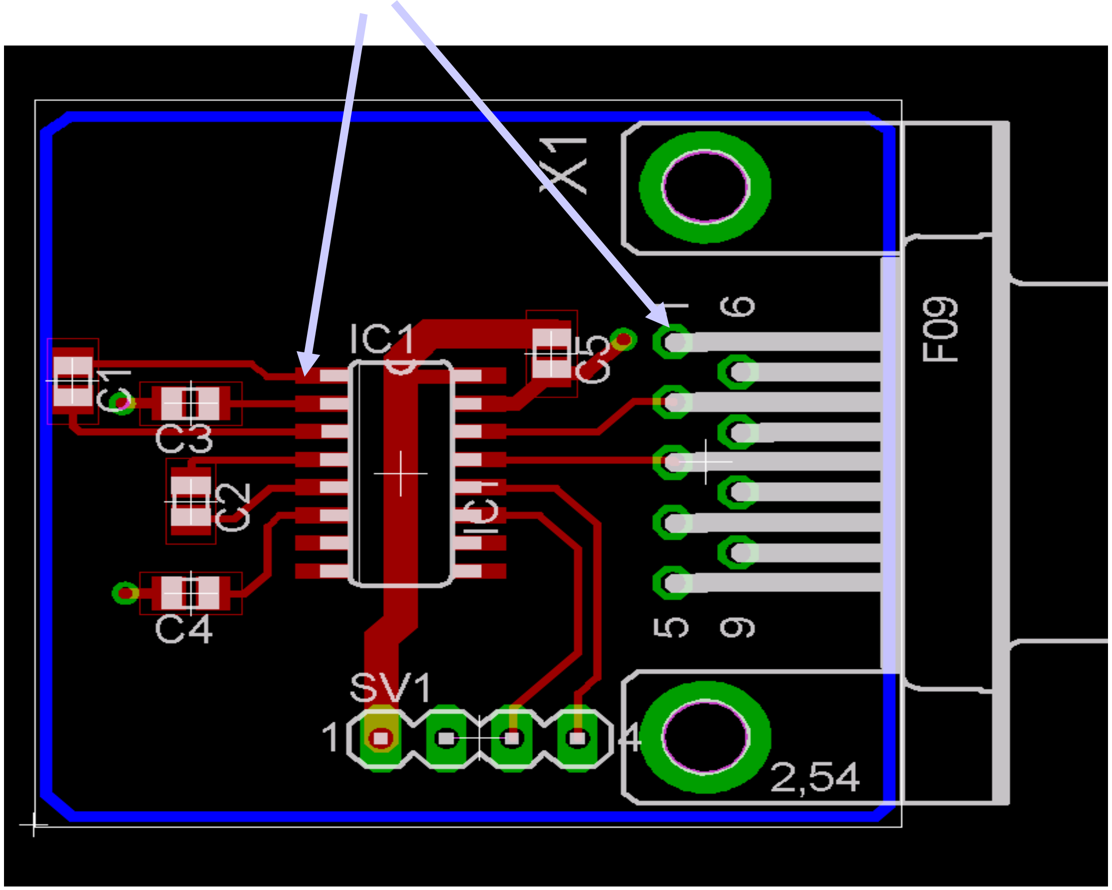
Item	Technical Parameters	Specification
Min Line Width mil	3	Finished copper thickness 0.5 OZ
Min Space mil	3	Finished copper thickness 0.5 OZ

PCB design

- Tracks
  - track width will be dictated by the current flowing through it, and the maximum temperature rise of the track. The thickness of the copper will also play a part, as will any solder coating finish.
  - thickness of the copper on the PCB is nominally specified in ounces per square foot.
  - the bigger the track width, the better. Bigger tracks have lower resistance, lower inductance, can be easier and cheaper for the manufacturer to etch, and are easier to inspect and rework.

  - Calculator programs exist (ex: www.ultracad.com/calc.htm) to figure out a required track width based on the current and the maximum temperature rise.

PCB design



Pads

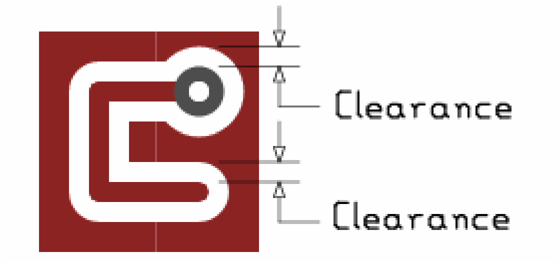
PCB design

- Pads
  - portion of copper on printed circuits to mount or attach components
  - component libraries define pad sizes
  - usually circular or oval
  - rectangular for pin 1 (to be different)
  - rectangular for surface mount.

- Vias
  - plated through hole to connect tracks between different layers of a PCB circuit board.
  - similar to but smaller than pads.
  - differently handled CAD software packages

PCB design

- Clearance
  - Clearance is the shortest distance between two conductive parts (isolating gaps)



* Figure from: http://www.delorie.com/pcb/docs/gs/gs.html

- Polygon
  - areas to automatically fill a desired area with copper.
  - useful for ground planes
  - solid or hatched

References

[1] David L. Jones, PCB Design Tutorial, Revision A, June, 2004, Available: www.alternatezone.com

[2] Kraig Mitzner, Complete PCB Design Using OrCad Capture and Layout, Elsevier, 2007

[3] Motorola Ships Passive-Embedded PCB for Mobile Phones,
May 2003, http://techon.nikkeibp.co.jp/NEA/archive/200305/244638/

- www.expressPCB.com
- www.4PCB.com
- www.pcbcore.come
- www.pcb-pool.com
-